View metadata, citation and similar papers at core.ac.uk

brought to you by CORE

provided by Missouri University of Science and Technology (Missouri S&T): Scholars' Mine

Missouri University of Science and Technology

Scholars' Mine

International Specialty Conference on Cold-Formed Steel Structures

(2006) - 18th International Specialty Conference on Cold-Formed Steel Structures

Oct 26th, 12:00 AM

Finite Element Analysis of Web Crippling Behaviour of Cold-formed Steel

M. Macdonald

J. Rhodes

Manoj A. Heiyantuduwa

Follow this and additional works at: https://scholarsmine.mst.edu/isccss

Part of the Structural Engineering Commons

Recommended Citation

Macdonald, M.; Rhodes, J.; and Heiyantuduwa, Manoj A., "Finite Element Analysis of Web Crippling Behaviour of Cold-formed Steel" (2006). *International Specialty Conference on Cold-Formed Steel Structures*. 3.
https://scholarsmine.mst.edu/isccss/18iccfss/18iccfss-session2/3

This Article - Conference proceedings is brought to you for free and open access by Scholars' Mine. It has been accepted for inclusion in International Specialty Conference on Cold-Formed Steel Structures by an authorized administrator of Scholars' Mine. This work is protected by U. S. Copyright Law. Unauthorized use including reproduction for redistribution requires the permission of the copyright holder. For more information, please contact scholarsmine@mst.edu.

Eighteenth International Specialty Conference on Cold-Formed Steel Structures
Orlando, Florida, U.S.A, October 26 & 27, 2006

FINITE ELEMENT ANALYSIS OF WEB CRIPPLING BEHAVIOUR OF COLD-FORMED STEEL FLEXURAL MEMBERS

M.Macdonald[1], M.A.Heiyantuduwa[1] and J.Rhodes[2]

ABSTRACT

Finite element analysis can be effectively used to investigate the complex failure behaviour of structural members. This paper illustrates the use of finite element analysis to model the web crippling behaviour of cold-formed steel flexural members and to investigate the influence of various geometric parameters on the web crippling strength. The interior-One-Flange loading condition was considered here and the fixed flange loading method was used to represent the actual loading cases. Non-linear finite element techniques were used to represent the large deformations, large rotations and the non-linear stress-strain characteristics. In this research, channel sections were used to investigate the web crippling problem and the results were compared and validated against the experimental results.

1. INTRODUCTION

When thin-walled cold-formed flexural members are subjected to concentrated loading or reactions, they may undergo localised failure, generally leading to overall collapse. This failure behaviour is defined as web crippling and is shown in Figure 1. Web crippling behaviour has been studied by various researchers using experimental, numerical and theoretical techniques for beam members, sheeting and deck sections. Most research studies have focused on improving design specifications with a few investigations carried out in order to get a better understanding of the failure behaviour. It has been identified that the empirical nature of current design specifications can make them of limited applicability and

[1] School of Engineering, Science & Design, Glasgow Caledonian University, Glasgow, UK
[2] Department of Mechanical Engineering, University of Strathclyde, Glasgow, UK

can only be used within the range of specimens tested. A research study was initiated with the intention of investigating web crippling behaviour using experimental, numerical and theoretical techniques. This paper presents the finite element modelling and analysis of the web crippling problem particularly for lipped channel section beams under Interior-one-flange loading.

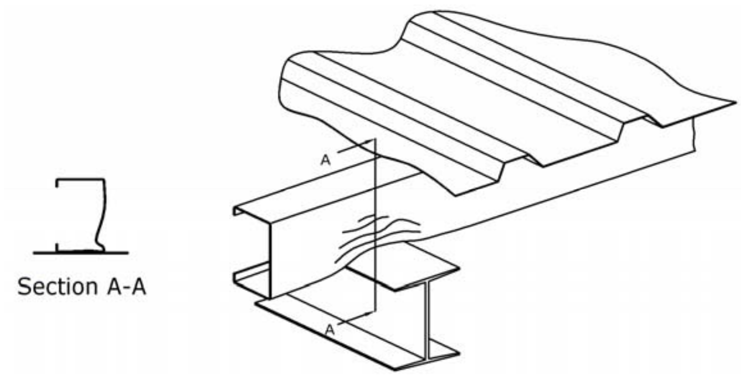

Figure 1: Web crippling failure at a support point [1]

The effect of the cold forming process on web crippling behaviour where at formed corners, increased work hardening increases the overall cross-section properties in terms of yield and ultimate strengths is also investigated.

Web crippling can take place under four different loading conditions; these are defined by the American Iron and Steel Institute (AISI) based on the position of the failure and the orientation of the loading and reactions [2]. Figure 2 shows these four basic loading conditions defined by AISI.

The objective of the finite element analysis of web crippling behaviour is to develop numerical models which can be used to represent the actual failure behaviour and to predict the ultimate web crippling load. Further, these models can be used to obtain an understanding of the post-failure behaviour of the members subjected to web crippling. Modern nonlinear finite element techniques provide a means to examine complex failure behaviour including large deformations, large rotations and nonlinear material characteristics. Loading conditions can be modeled using contact elements or nodal forces depending on the mode of loading. Often half or quarter models are developed based on the symmetry of the member and the loading condition.

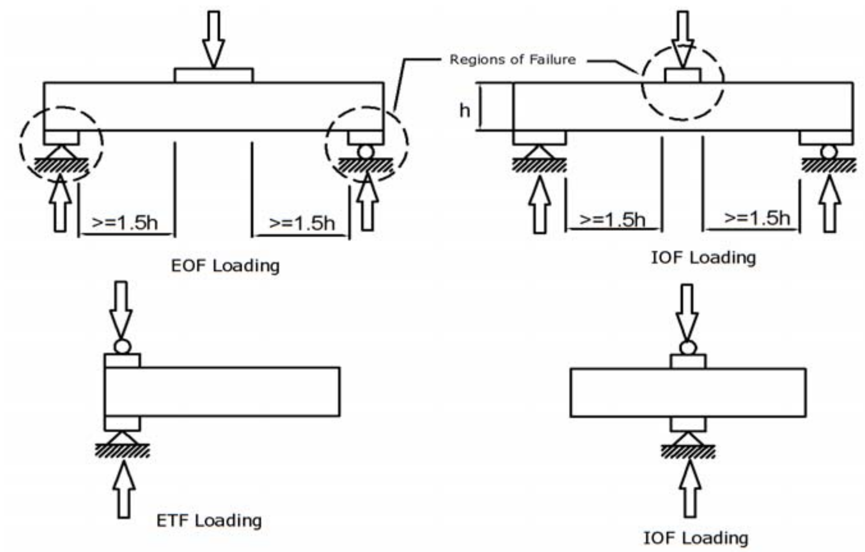

Figure 2: Loading conditions of web crippling tests [2]

In most cases finite element models can be used as a substitute for expensive and time consuming experimental programs, and dimensional parameters can be easily changed to obtain a wide range of test specimens. Further, these models can be used to test a wide range of assumptions, and simulate experiments that are practically impossible to perform but worthy of studying [3, 4].

The reliability of the model and the results depends on a large number of factors. Initially the model itself should be able to represent the actual geometry of the member and the elements should be able represent the physical properties of the actual material as close as possible. The generation of the element mesh is an important process, as in most cases the accuracy of the results largely depends on the mesh. Loading and boundary conditions must also accurately represent the actual loading and boundary situation.

2. FINITE ELEMENT MODEL DEVELOPMENT

In 1989, Sivakumaran carried out a finite element analysis to investigate web crippling behaviour and to determine ultimate web crippling strength of lipped channel sections [5], where the analysis extended up to ultimate load levels. The following points were highlighted when selecting the proper finite elements: steel lipped channel sections were considered with very thin webs, flanges and lips, and were subjected to in-plane and bending actions. Hence finite elements were able to represent membrane behaviour as well as flexural behaviour. In this study, the finite element package called ADINA was used, with the degenerated isoparametric shell element employed to analyse the problem. Half of the specimen was considered for the mesh generation because of the symmetry of the loading. In order to capture the local large deflection and local yielding in the region around the web crippling area, finer meshing was considered in this region. Further away from the loaded area, larger elements were used. Results of the finite element analysis showed that the general deformation shape was similar to the experimental deflected shape and predicted ultimate loads which were well in agreement with the experimental values within a 9% tolerance.

A number of research studies have been carried out using finite element analysis to model web crippling behaviour of sheet and deck sections. In 2000, Hofmeyer [3, 6, 7] carried out a finite element analysis program to study the post-failure behaviour of deck sections. Three different finite element models were developed to investigate the post-failure mechanisms namely, rolling, yield arc and yield eye post-failure mechanisms. The three models were different in terms of mesh, symmetry and loading. The experimental and simulation results were compared for: load vs. beam deflection, load vs. support rotation and, load vs. web crippling deformation and good agreement was obtained.

3. FINITE ELEMENT ANALYSIS PROCEDURE

The finite element analysis procedure basically comprises of three different stages: development of the geometric model and obtaining the finite element model, solving the system, and obtaining and viewing the results. The accuracy of the results mainly depends on the finite element model and the boundary conditions. The finite element model and the boundary conditions should be able to represent the actual member and the loading condition in order to get the results which are much closer to the real situation. The finite element package used in this research is *ANSYS* (Release 8.1- University Advanced Version).

The various steps in the finite element procedure are described below with respect to the analysis of web crippling behaviour of cold-formed steel lipped channels.

3.1. Geometric model design

The first step is the modelling of the geometry involving the definition of the cross-section using key points, lines and arcs. The corner radii were modelled using two arcs per corner. The half member was obtained by dragging the cross section along the length of the member.

The support reaction block which is shown in Figure 3, was modelled as a beam using beam elements. The beam was merged to the section for the purpose of meshing.

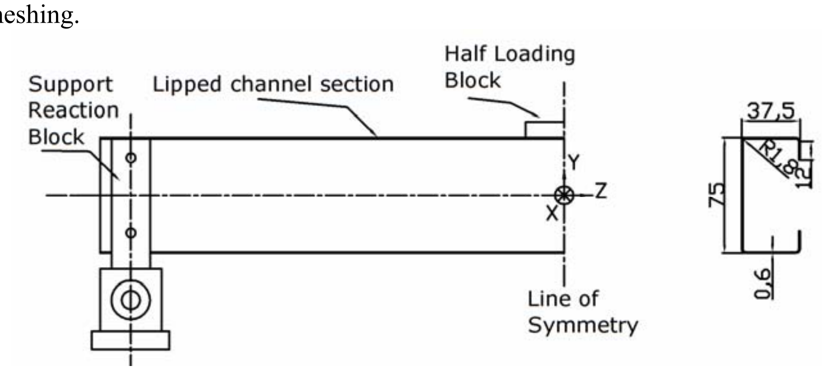

Figure 3: Geometry of loading and channel cross-section

3.2. Finite element model: Mesh and Elements

Four different meshed regions were used in the model based on mesh sizes. Under and near the load bearing plate where the large stress variation occur and failure initiates, a small element size was used, further away from the load bearing plate a relatively large mesh size was used, between the fine and coarse meshed regions another region was defined for smooth transition from fine mesh to coarse mesh. Line meshing method was used as this allowed the variation of mesh sizes within a given area.

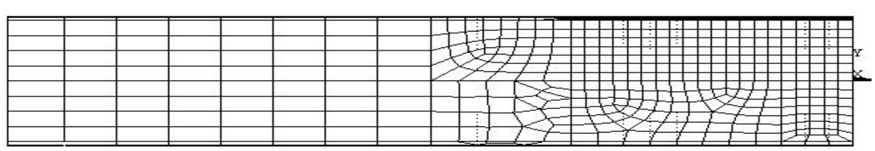

Figure 4(a): Element mesh -Side view

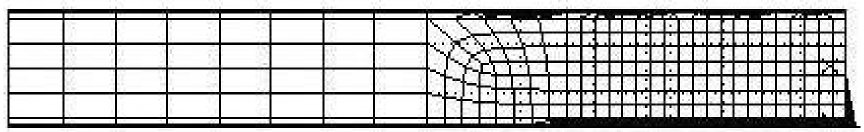

Figure 4(b): Element mesh -Top view

From the many ANSYS elements, Shell43 was used to model the channel sections and Beam4 was used to model the supports. Shell43 is a four noded element which has plasticity, large deflection and large strain capabilities [8].

3.3. Material properties

Material properties were obtained by carrying out a series of tensile tests. Elastic modulus and Poisson's ratio were used as the basic material properties. Stress vs. strain curves were obtained directly from the tensile tests and converted to the real stress vs. real strain curves. The full section stress vs. strain curve was represented using eight distinct points from the curve and these points were fed into ANSYS using a material data table. This enabled the type of plasticity model to be defined as Multi-linear isotropic hardening. The thickness of the zinc coating was neglected as it was negligible compared to the core thickness of the steel and the element thickness was taken as the overall thickness of the sheet metal.

3.4. Loading and boundary conditions

Load was applied as nodal displacements under the load bearing plate, where the nodes were coupled together to represent the fixed condition of the flange to the load bearing plate. The symmetric boundary conditions were applied at the end of the member, Z-direction displacement was fixed and the rotations about the X-axis and Y-axis were also fixed. The support boundary conditions were defined to represent the actual support blocks used in the experimental setup. As mentioned earlier, the support block was modelled as a beam member attached to the section, and the boundary conditions were applied to the bottom end of the support beam. At this point all displacements and rotations were restricted except X-direction rotation.

4. BASIC LINEAR ELASTIC MODELS FOR INITIAL STUDY

Initial finite element models were developed and analysed without considering material and geometric nonlinearities in order to check the accuracy of the geometry, mesh, and the boundary and loading conditions. These are basic elastic models and only used to optimise the number of elements used within a particular meshed area and between different meshed regions. However, these models are capable of providing a good general understanding of the behaviour within the elastic range and up to the ultimate load limit.

Load was applied as nodal forces and a coupled degree-of-freedom was used to represent the fixed-flange condition on the load bearing plate. Mesh density was varied at the top corner radius, web and the top flange in order to find out the optimum sizes of mesh required under the load bearing plate, near the load bearing plate and somewhat away from the load bearing plate. As the number of elements were changed in various meshed areas, the stress, strain and deformation values were changed according to this and converged to a certain value. The two graphs shown in Figure 5(a) and 5(b) indicate that there should be a minimum of six elements to successfully model the corner radii under and near the load bearing plate. This process was continued for all other areas and compared the stress, strain and displacement against the mesh sizes or the number of elements used.

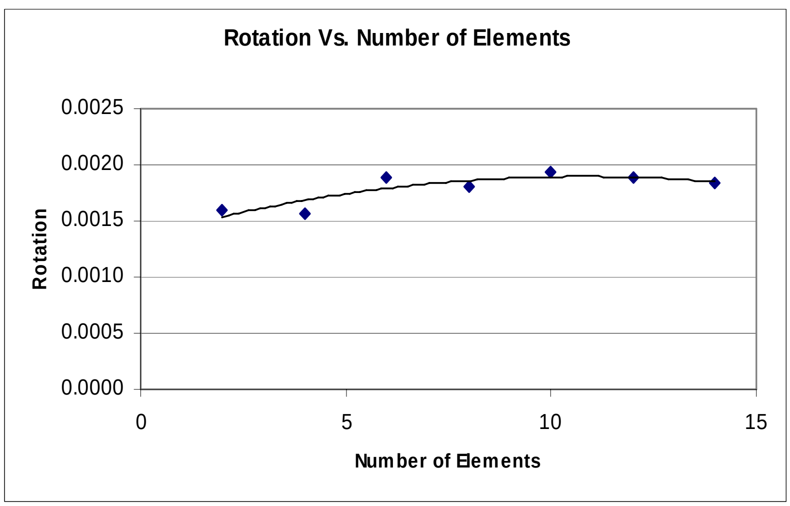

Figure 5 (a): Rotation of the reaction at the support point vs. number of elements in the top corner radius

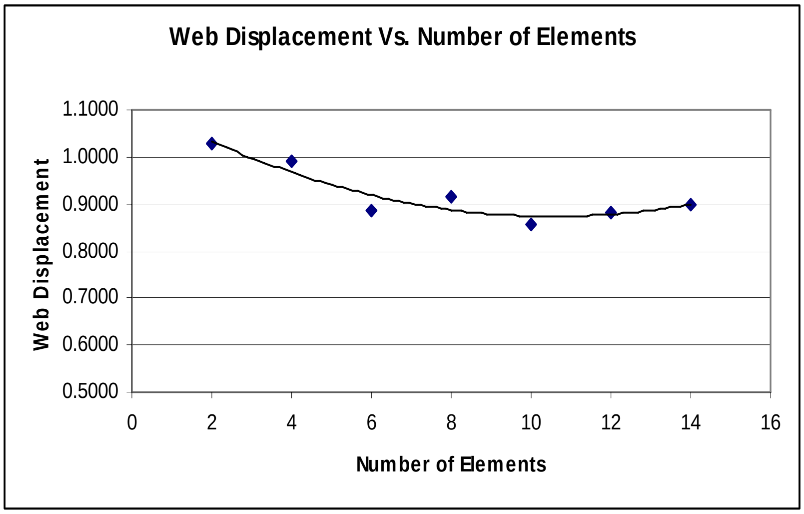

Figure 5 (b): Transverse web displacement vs. number of elements in the top corner radius

There are a few other methods which can be used to optimise the mesh and the number of elements. For example, it is possible to use some experiments where a test specimen is strain gauged to measure the strain values with LVDTs to capture small displacements. These strains and displacements can be compared with the FE elastic analysis results to find out the optimum number of elements in the predefined areas.

5. NONLINEAR MODELS TO REPRESENT COLLAPSE BEHAVIOUR

These models include both material and geometric nonlinearities. Material nonlinearity is used to model the post-failure (collapse) behaviour. The material properties comprise of elastic modulus, Poisson's ratio and the data table containing the points related to the stress-strain curve. Geometric nonlinearities mainly comprise of large deformations, rotations and contact surfaces. For an initial analysis, the loading was applied using a loading block which was fixed to the top flange of the specimen tested. This enabled us to model the loading condition using nodal displacements and the coupled degree-of-freedom. Further, displacement control was employed for the loading of the beam and a number of load steps were used to gradually increase the displacement of the loading. ANSYS basically uses two approaches to solve nonlinear problems, namely, "Newton-Raphson" method and Arc-length method. Here the "Newton-Raphson" method was employed [8].

Results of the finite element analysis carried out on the 75mm web height lipped channel sections for IOF loading condition is given in this paper. Altogether nine models were developed to represent various section parameters: Corner radii (1.25mm, 2.5mm and 4.5mm) for consideration of the effects of cold forming, and bearing lengths (25mm, 50mm and 100mm).

6. RESULTS

Results of the analysis were obtained by using general postprocessor /POST1 and time history postprocessor /POST26 which are built into the ANSYS software. General postprocessor /POST1 was used to view the results of the analysis and to obtain the deformed shape and the various nodal stress and strain values. /POST26 provides the results over the time history and was very useful in obtaining the load vs. deformation graphs under load bearing plate and other critical locations [8].
The load was calculated against the displacement of the load, and graphs were obtained. Additionally, reaction force at the support point, rotation at the support and the transverse deflection of the web were plotted against the displacement of the load. Figure 6 shows the deformed shape of the channel section of 75mm web height with load bearing length 50 mm and corner radii 4.5mm.

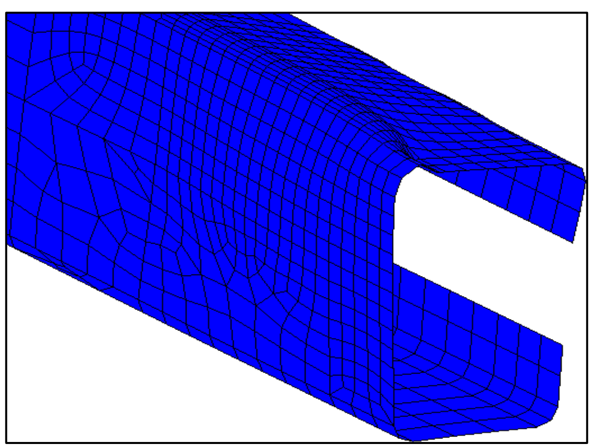

Figure 6: Deformed shape

Figure 7 shows the load vs. load point displacement curves for three different models with 75mm web height, 50mm bearing length. The difference between the three models is that they have three different corner radii values: 1.25mm, 2.5mm and 4.5mm.

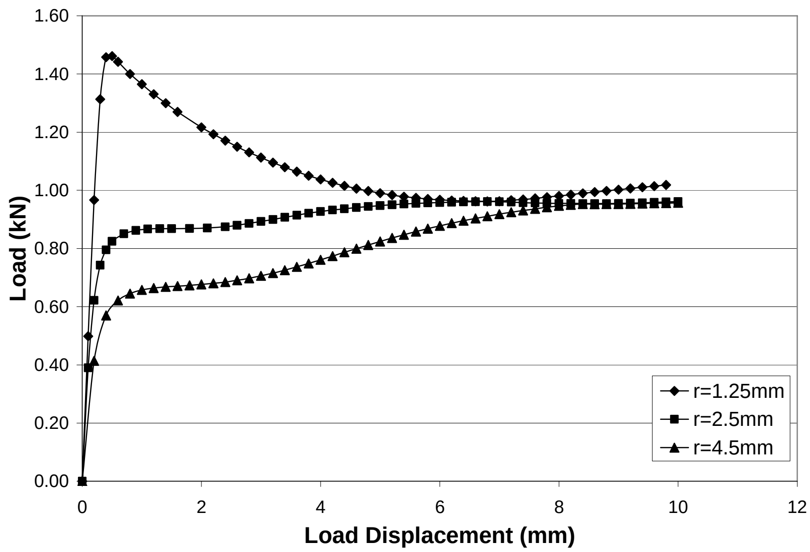

Figure 7: Load vs. load point displacement for three different corner radii: 1.25mm, 2.5mm and 4.5mm and bearing length 50mm

7. COMPARISON OF FE RESULTS AND EXPERIMENTAL RESULTS

Results of the FE analysis and experimental results were compared in order to attempt to validate the FE models. The results for the 75mm web height and 50mm bearing length test series are compared and are shown in Figures 8, 9 and 10 respectively. In Figure 8, where the corner radius value is 1.25mm, the FE load prediction is significantly higher than the experimental results. Reasons for this could be due to the small corner radius at the top flange and web interaction, providing very high stiffness to the member in the FE model, whereas in the experiments the actual beam webs have imperfections, leading to lower values of ultimate strength. Even though the ultimate strength values show a significant difference, the two curves follow the same deformation shape which indicates that the failure mechanism was predicted reasonably well in the FE analysis.

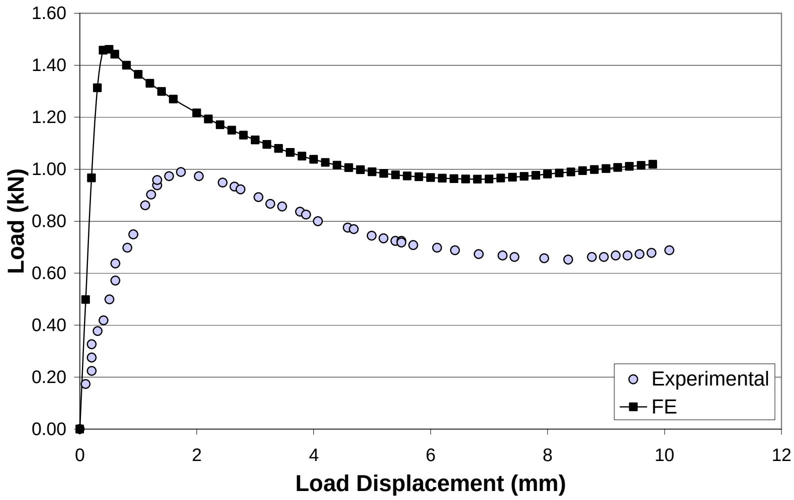

Figure 8: Load vs. displacement curves obtained from experimental and finite element studies for corner radii 1.25mm and bearing length 50mm

Figures 9 and 10 show the curves for the specimens with corner radii values 2.5mm and 4.5mm respectively, the ultimate web crippling strength predicted by the FE analysis is less than the experimental results. The reason for this could be due to the strength increase in the corner radii due to the additional work hardening gained during the cold forming process. This provides a relatively higher strength not only near the corner radii but also throughout the section as well.

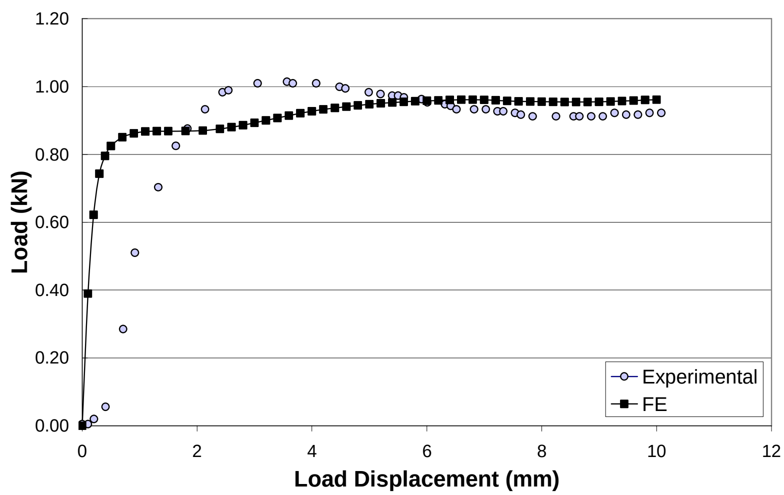

Figure 9: Load vs. displacement curves obtained from experimental and finite element studies for corner radii 2.5mm and bearing length 50mm

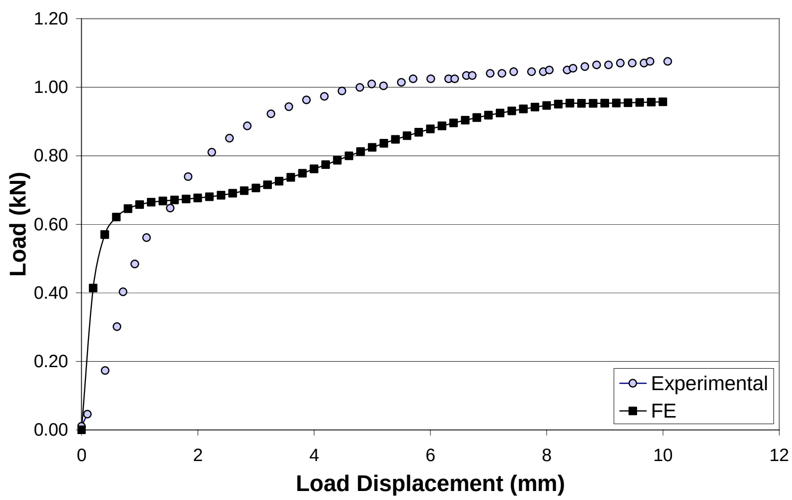

Figure 10: Load vs. displacement curves obtained from experimental and finite element studies for corner radii 4.5mm and bearing length 50mm

8. CONCLUSIONS

Finite element models were developed to simulate web crippling failure of cold-formed lipped channel section beams under interior-one-flange loading condition. Section parameters and the loading were varied in order to have a wide range of test specimens for the experimental program. This enabled an attempt to be made to compare the experimental results with FE analysis. ANSYS finite element package was used for the analysis and nonlinear techniques were employed in order to represent the plastic failure in the post-yield region.

The load vs. load displacement curves obtained from the FE study and the experimental study showed that the FE analysis was able to capture the failure mechanisms even though the ultimate strength values were substantially different. Reasons were identified as possibly being the imperfections in the members and the strength increase due to cold-working (work-hardening) during the manufacturing process.

Further, this research is intended to continue the analysis of the other loading situations, and extended to represent the imperfections and use of contact element for loading. It is intended that the of modelling of the member will also include the variation of the material properties across the cross-sections by including the strength increase due to cold working of the manufacturing process.

REFERENCES

1. J. Rhodes and D. Nash, **"An investigation of web crushing behaviour in thin-walled beams".** *Thin-Walled Structures,* vol. 32, pp. 207-230, 1998.
2. W.W. Yu, "**Cold-Formed Steel Design".** John Willey & Sons, Inc, 2000.
3. H. Hofmeyer, **"Combined Web Crippling and Bending Moment Failure of First-Generation Trapezoidal Steel Sheeting".** *Ph.D. Thesis,* 2000.
4. O. Kaitila, **"Web Crippling of Cold-Formed Thin-Walled Steel Cassettes".** *Ph.D. Thesis,* 2004.
5. K. S. Sivakumaran, **"Analysis for Web Crippling Behaviour of Cold-Formed Steel Members".** *Computers and Structures,* vol. 32, pp. 707-719, 1989.

6. H. Hofmeyer, J. Kerstens, B. Snijder and M. Bakker, **"FE Models for sheeting under interaction load".** *15th International Speciality Conference on Cold-Formed Steel Structures,* St. Louis, Missouri U.S.A., pp. 105-119, Oct 19-20 2000.
7. H. Hofmeyer, J. Kerstens, B. Snijder and M. Bakker, **"New prediction model for failure of steel sheeting subjected to concentrated load (web crippling) and bending".** *Thin-Walled Structures,* vol. 39, pp. 773-796, 2001.
8. ANSYS Release 8.1 (2004), User's Manuals, ANSYS, Inc.